SIEMENS
Ingenuity for life

Siemens Digital Industries Software

What they didn’t teach you in school about mixing

Executive summary

Ensuring adequate chemical mixing is a complicated matter. The act of mixing dilutes properties held by each component and may even cause chemical reactions. Additionally, how chemicals mix is affected by factors such as temperature, material properties, pressure, and flow rate. Inadequate understanding of the above fluid flow factors combined with the equipment used potentially can negatively affect mixing results and lower product quality while increasing costs to fix problems down the road.

Therefore, design engineers need to be able to verify that their models accurately predict the chemistry and physics of the mixing process. While you can use the tried and tested method of creating a prototype for each idea, it is much faster and less expensive to use simulation software.

Simulation software such as computational fluid dynamics (CFD) use a set of equations called Navier-Stokes to represent fluid flows because they are very good at representing and evaluating complex flow patterns. As a result, CFD solutions are fast and efficient for exploring designs and providing proof of concept.

Contents

Simulation is a great mixture problems solver

Among the typical questions that simulation software can answer are:

- Has the proposed design fixed the ratio and proportion of the mixture?
- Does the model have the right flow pattern?
- What kind of mixing is ideal for my application: continuous, batch or turbulent?
- Should we use a pump or rely on flow?
- Will the viscosity change during system operation under different conditions and can the equipment handle the change?
- Which impeller works best for getting the best mix ratio?
- Which motor size is the most appropriate one for my application?

Simulation software can answer these and more questions while giving you valuable insight into your proposed designs easily, quickly and cost-effectively.

Best ROI is achieved by simulating early and often

CFD software traditionally can be complex and difficult to master, so its use has been relegated to the final-prototype testing phase and given over to CFD experts or specialists. However, testing a design only at the prototype stage is costly. According to a report by Lifecycle Insights [1], at this stage, failed designs lead to missing project milestones, extra rounds of testing, and having to work long hours. Multiple surveys conducted by various industry analysts and CAE vendors suggest that the most successful companies assess performance of their designs early during the development process and actively promote collaboration and sharing of knowledge between analysis experts and design engineers.

Reducing the cost of change and building in more room for cost reductions provides the biggest return on investment (figure 1) [2]. Prof. Martin Eigner coined "frontloading" as an umbrella term for the practice of using a whole slew of software simulation tools, including CFD, earlier in the design process [2].

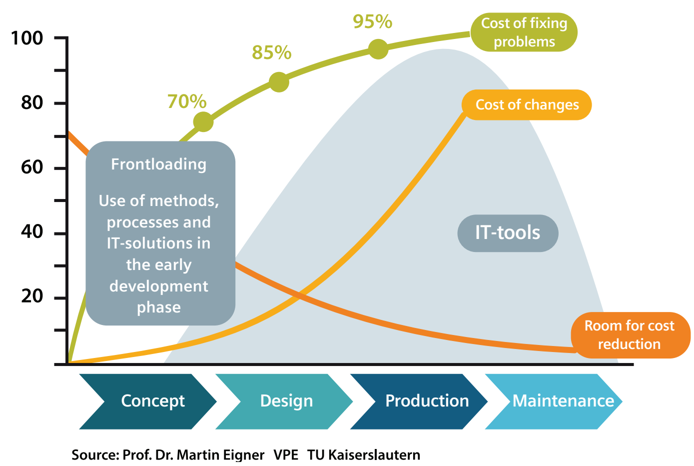


Figure 1: Frontloading simulation can help reduce costs.

How frontloading CFD has changed the design process

About 20 years ago, stress analysis was introduced for use during the early design stages, and it quickly became an integral step in the development process. Now, all major MCAD software tools provide design-level stress simulation. However, frontloading stress simulation and conducting analysis early during the design stage did not mean that manufacturers stopped simulating during the validation stage. Simulation simply became a method by which trends were examined and less desirable design ideas were dismissed.

Unlike the verification stage, during the design phase, speed is of the essence. Engineers need to simulate, not only early, but often, to keep in step with the speed of design changes. By iterating rapidly, engineers can discard the less attractive ideas and innovate further. Once a design has been fully explored and identified as viable, it can continue on to the verification stage.

This early virtual-prototyping practice has spread to other areas including CFD analysis. We now have CFD tools that are designer-friendly and that are linked integrally and conveniently within CAD tools. Using these combined tools, a digital twin, a virtual representation of the product, can be created quickly and inexpensively.

The benefits of frontloading CAD-embedded CFD include:

- Better matching to product requirements (for example, lower weight, faster speed, etc.)
- Reducing downstream development delays and costs (such as reduce testing and prototyping, reduce change orders, etc.)
- Satisfying customer contractual obligations or regulatory requirements
- Reducing product lifecycle costs
- Driving production costs lower
- Getting products to market faster

Why designers need CFD for simulation-driven design

Using the right tool at the right time is key to success. The main task of a design engineer is design. As such, your CAD software is your primary tool and to ensure your design and your simulation data does not diverge, your CAD data should be at the center of all your simulation activity. By using CAD-embedded simulation, it becomes an add-on functionality inside your CAD. CAD-embedded CFD uses native geometry for analysis; therefore, exporting data and healing it in preparation for analysis is not required. The software simply slots in, without the pain of having to learn how to work with a new interface. CFD simulation becomes simply another functionality offered by the CAD package.

However, simply embedding CFD in CAD is not enough. You also need intelligent automation. CAD-embedded CFD programs feature built-in intelligent automation for easier, faster, and more accurate analysis. For example, in mixing, a designer is interested in understanding what is happening in the negative space, the empty space where the fluid resides. Simulation-driven design CFD solutions are intelligent enough to recognize that the empty space is the fluid domain so time is not wasted on creating geometry to accommodate software.

Also, being able to handle turbulence models effectively and efficiently is important for mixing simulation. To ensure you are getting the correct turbulence model, the flow features likely to be present in the application need to match with the models available in the simulation solver. However, there is an abundance of turbulence models so knowing when a flow goes from laminar to turbulent is important and requires years of specialist knowledge. After that, knowing which turbulent model to use is the next challenge. Simulation-driven, designer-friendly CFD solves this issue by incorporating algorithms that automatically reflect the right flow during calculation.

In addition, before analysis can begin, the model has to be meshed – but how do you know which meshing method best depicts the flow phenomenon. Designer-friendly CFD uses a fully automated mesher that will generate the best possible mesh for the problem.

Lastly, a CFD solution for design must be fast to keep up – after all, who can afford to wait days, if not weeks, before knowing whether a proposed model shows promise? Design engineers need answers in a timely manner to ensure a timely delivery of the product under design.

Mixing problems solved

Simcenter FLOEFD™ by Siemens Digital Industries Software is such a CFD solution. Created for designers, it is embedded in the most popular MCAD toolsets such as CATIA™ V5, Creo® Elements/Pro™, Siemens NX™ and Solid Edge®. It provides a general-purpose CFD tool that designers can use to solve many engineering problems, including ratio and proportion mixture problems while working on their design. It runs completely inside CAD, providing solutions from solid modeling to problem setup, solving, visualizing results, optimizing the design, and reporting.

With Simcenter FLOEFD, designers can focus on solving a wide range of mixing-related reaction forces problems, such as multiple-species transport, conjugated heat transfer between fluids and surrounding solid materials, and flow through multiple inlets and/or outlets with various fluid concentrations. What-if scenarios can be run to analyze complex physical processes in the mixing simulation, including heat transfer, mixing of various species of gases or liquids, and non-Newtonian fluids. It can then be used to quickly modify and optimize the design's geometry inside MCAD (figure 2).

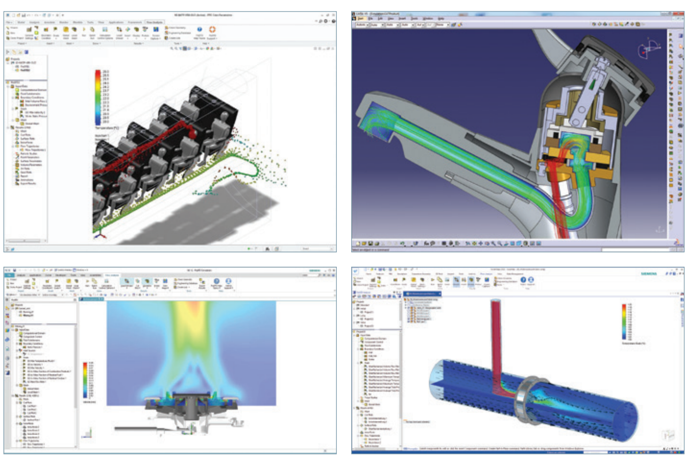


Figure 2: Simcenter FLOEFD is embedded into popular MCAD programs.

Simcenter FLOEFD solves for the species diffusivities as a function of pressure, temperature and species concentration in fluid mixtures, which is why it can be used to analyze many different applications. Typical mixing applications include static and dynamic mixers, mixing of different fluids of the same phase, and mixing of different temperature fluids. For example, a designer can look at the mass fraction distribution and temperatures of the mixing fluids in a dynamic or static mixer while also using the tool to predict the pressure load onto the mixing geometry for structural analysis later. Combining these parameters into a single model helps design a better product earlier.

Because Simcenter FLOEFD software is CAD-embedded, all that is needed to use it during design development is installing in the MCAD system and applying the physics of the product. All the menus and commands necessary to run a full CFD flow analysis are installed into the CAD menus for you. Most designers can start using Simcenter FLOEFD on their designs after eight hours of training.

The starting point of any fluid mixing analysis is to define the overall boundary conditions of the problem. Simcenter FLOEFD has a wizard that walks users through the setup, including selection of individual fluids. Simcenter FLOEFD can use existing MCAD models for analysis, without having to export or import additional geometry. The embedded Simcenter FLOEFD toolset can use newly created or existing 3D CAD geometry and solid model information to simulate designs in real-world conditions.

Once a project is created and the boundary conditions applied, the model needs to be meshed, that is, a computational grid has to be built. Simcenter FLOEFD creates meshes automatically in minutes. CFD-embedded in the CAD program creates an adaptive mesh that reduces the cell size where necessary, increasing the resolution of the analysis, to ensure more accurate simulation results in complex areas of the model (figure 3).

Once you have optimized the design for flow, then you can move to structural analysis software such as Simcenter 3D for vibration analysis.

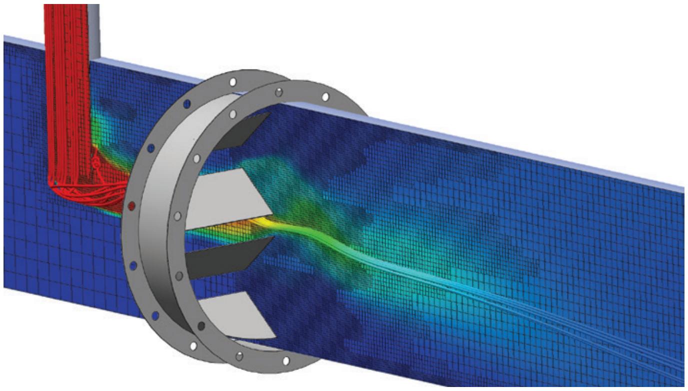


Figure 3: Simcenter FLOEFD SmartCells technology provides accurate analysis results.

How Simcenter FLOEFD solves ratio and proportion mixture problems

When analyzing fluid mixing processes, building a mesh is important to capture the complex geometry of the system or device as well as the fluid flow gradients. The mesh is simple in concept, yet it is the heart of complex CFD calculations. The surface of the device is mapped into tiny rectangular cells, each of which is split into solid and fluid volumes that are analyzed discretely. The process then develops a composite result that incorporates all of the cells. Simcenter FLOEFD enables visualizing what is happening to fluid concentrations, providing valuable insight to guide decisions so the design can be interrogated more thoroughly for better ratio and proportion mixture results.

One way to examine the mixture mass fraction is to use a cut plot, which depicts the mass fraction distribution of a specific fluid on a plane through the model (figure 4). A cut plot of results can be displayed with any results parameter, and the representation can be created as a contour plot, isolines or vectors. The cut plot also can be created using any combination, such as velocity magnitude and vectors. In addition to cut plots, an isosurface plot can be displayed easily to show regions of a certain mixture ratio. This can help to spot remaining pockets of high fluid concentrations that were not mixed adequately in the process and require a design change to improve the efficiency of the mixer.

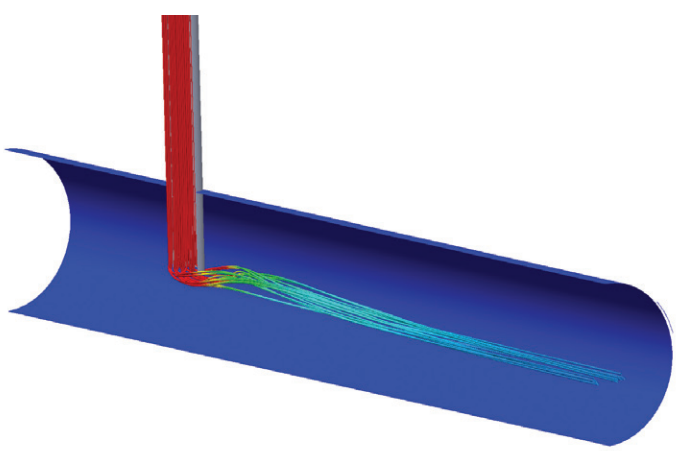


Figure 4: Cut plot.

Solving ratio and proportion mixture problems is an iterative process. After seeing the initial analysis results, most designers want to modify their models to explore different scenarios. Simcenter FLOEFD makes it easy to conduct these what-if analyses. Design alternatives can be explored, design flaws detected, and product performance optimized before detailed designs or physical prototypes are created. This allows quick and easy determination of which designs have promise and are unlikely to be successful.

Multiple clones of the simulation projects can be made in Simcenter FLOEFD that automatically retain all analysis data such as boundary conditions for different variations of the geometry. When modifying a solid model, it can be analyzed immediately without having to reapply boundary conditions and material properties.

Simcenter FLOEFD software operates immediately on the changed geometry, creating a new mesh automatically while working within the previously defined boundary conditions. Thus, the step from a changed geometry to running the solver and examining results is greatly accelerated. Its compare configuration and parametric study capability enables designers to understand the influence of changes in the geometry or boundary conditions on the results.

You can evaluate the design envelope by assessing results using numerical values, graphs, and/or visual images/animation to compare a wide range of project permutations. This is how Simcenter FLOEFD accelerates the iterative design process, by enabling designers to quickly and easily incorporate knowledge gained in an analysis to improve the product.

Simcenter FLOEFD provides robust verification capabilities for validating designs. Before releasing a new version of Simcenter FLOEFD, Siemens Digital Industries Software engineers validate the release with a suite of 300 tests. Based on this rigorous verification suite, Simcenter FLOEFD offers 20 tutorial and 32 validation examples, including their documentation, ready for immediate use.

Sharing results and findings is simple. Simcenter FLOEFD is fully integrated with Microsoft® Word® and Excel®, allowing engineers to create documents and collect important data in graphical form from any project for reports. In addition, it automatically creates Excel spreadsheets that summarize the outputs of an analysis; this makes the last step in any analysis, creating reports, effortless. Simcenter FLOEFD also comes with a free standalone viewer that can be used to share selected result plots with your customers in an interactive 3D environment rather than just a 2D image.

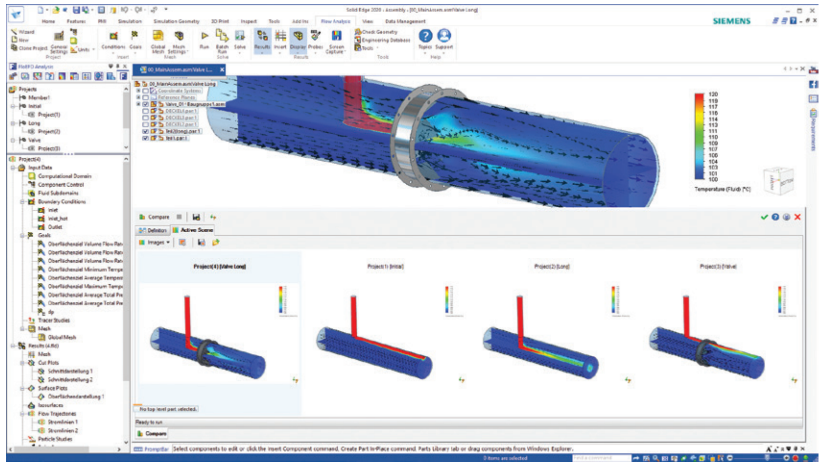
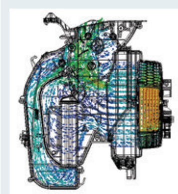
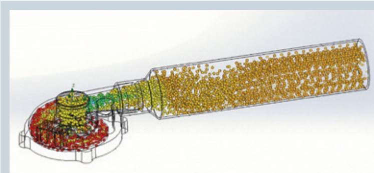
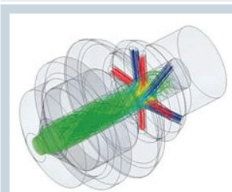
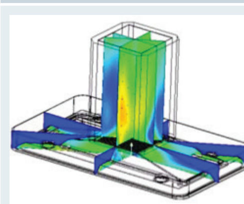
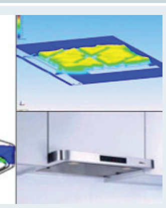


Figure 5: Simcenter FLOEFD parametric study and design comparison functionality helps engineers optimize designs quickly.

Because of its design-centric foundation, ease-of-use, intelligent technology and fast robust solver, Simcenter FLOEFD makes front-loading CFD simulation a reality, reducing analysis time significantly – some of our customers have reported up to 75% reduction in the overall simulation time and enhanced productivity by as much as 40x, simply by opening the world of simulation to the design team.

Real world designers and Simcenter FLOEFD

Take a look at these real-world examples that demonstrate the speed, accuracy and power of Simcenter FLOEFD in helping designers meet tight deadlines, achieve higher quality results, and keep costs to a minimum.

Mercury Racing	Learn about the challenges the design team faced when designing an intercooler filter on a sterndrive engine	
Pan Asia Technical Automotive Center	Check out how the team ensures a comfortable vehicle ride for passengers by optimizing an air handling unit for uniform temperatures	
Borusan R&D	Read how the largest manufacturer of engine valves in Turkey started a collaborative project with the goal of developing and manufacturing completely new turbochargers	
APEX Group	Discover how the group develop and manufacture high-performance, heavy-duty equipment for heat recovery and gas handling projects	
Graco	Explore how the team substantially improved the performance of their spray gun products while significantly reducing their time to market	
Marenco	Learn how the team needed only half the time in comparison to traditional engineering methods to design an extractor hood	 

Siemens Digital Industries Software

Headquarters
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 972 987 3000

Americas
Granite Park One
5800 Granite Parkway
Suite 600
Plano, TX 75024
USA
+1 314 264 8499

Europe
Stephenson House
Sir William Siemens Square
Frimley, Camberley
Surrey, GU16 8QD
+44 (0) 1276 413200

Asia-Pacific
Unit 901-902, 9/F
Tower B, Manulife Financial Centre
223-231 Wai Yip Street, Kwun Tong
Kowloon, Hong Kong
+852 2230 3333

About Siemens Digital Industries Software

Siemens Digital Industries Software is driving transformation to enable a digital enterprise where engineering, manufacturing and electronics design meet tomorrow. Our solutions help companies of all sizes create and leverage digital twins that provide organizations with new insights, opportunities and levels of automation to drive innovation. For more information on Siemens Digital Industries Software products and services, visit siemens.com/software or follow us on LinkedIn, Twitter, Facebook and Instagram. Siemens Digital Industries Software – Where today meets tomorrow.

siemens.com/software

81895-C2 4/20 H